A Brief Tutorial

For

"How to Setup HSIM and HSPICE at UVa ECE Environment"

Prepared by Arijit Banerjee (ab9ca@virginia.edu)

Revisions: 1, dated August 13, 2012

TABLE OF CONTENTS

Chapter 1

Creating Setup File for HSIM and HSPICE

In order to create the setup file which has to be sourced in a bash shell, copy the following lines in italics and create a text file named syn_steup in Unix prompt with the below content:

```
#!/bin/bash

## Author: Ben Melton (ben.melton@virginia.edu)

## Modified: Mircea Stan (mircea@virginia.edu)

## Modified: Stevo Bailey (sdb6t@virginia.edu)

## Modified: Arijit Banerjee (ab9ca@virginia.edu)

## Setup License

SNPSLMD_LICENSE_FILE=27284@license.ece.virginia.edu

if [ -z "$LM_LICENSE_FILE" ]

then

  LM_LICENSE_FILE=$SNPSLMD_LICENSE_FILE

  export LM_LICENSE_FILE

else

  LM_LICENSE_FILE=$SNPSLMD_LICENSE_FILE:$LM_LICENSE_FILE

  export LM_LICENSE_FILE

fi

unset LANG

unset LC_ALL

export SNPSLMD_LICENSE_FILE

## Thanks to Gary Li (garyli@virginia.edu)

export SYNOPSYS=/app/synopsys

export EDITOR=/usr/bin/vim
```

Setup Core Synthesis Tools

CS_HOME=$SYNOPSYS/CoreSynthesisTools/F-2011.12

export PATH=$PATH:$CS_HOME/bin

Setup Custom Designer

CD_HOME=$SYNOPSYS/customdesigner/F-2011.09-SP1-2

#export CDSHOME=/app/cadence/IC615

export PATH=$PATH:$CD_HOME/bin

Setup HSpice

HS_HOME=$SYNOPSYS/hspice/F-2011.09-SP1

export PATH=$PATH:$HS_HOME/hspice/bin

#export PATH=$PATH:$HS_HOME/hspice/amd64

Setup HSIM PLUS

HS_HOME=$SYNOPSYS/hsimplus/F-2011.09-SP1

export PATH=$PATH:$HS_HOME/hsimplus/bin

#export PATH=$PATH:$HS_HOME/hsimplus/platform/amd64/bin

Setup Custom Explorer

CE_HOME=$SYNOPSYS/customexplorer/F-2011.09-SP1

export PATH=$PATH:$CE_HOME/bin

Setup Cosmos Scope

COSS_HOME=$SYNOPSYS/CosmosScope/F-2011.09

export PATH=$PATH:$COSS_HOME/ai_bin

Setup Formality

FORM_HOME=$SYNOPSYS/formality/F-2011.12

export PATH=$PATH:$FORM_HOME/bin

Setup ICC

ICC_HOME=$SYNOPSYS/icc/F-2011.09-SP2-1

export PATH=$PATH:$ICC_HOME/bin

WARNING: This section changes the $SYNOPSYS variable to work correctly.

Comment the last line if other tools are not working.

Setup Module Compiler

MC_HOME=$SYNOPSYS/CoreSynthesisTools/F-2011.12

export PATH=$PATH:$MC_HOME/amd64/mc/bin:$MC_Home/amd64/syn/bin

export MCDIR=$MC_HOME/mc

export MCENVDIR=$MCDIR/localadm:$MCDIR/adm

export MCLIBDIR=$SYNOPSYS/pdk/SAED_EDK90nm/Digital_Standard_cell_Library/synopsys/models

export SYNOPSYS=$MC_HOME

After doing so, write the file, and store in your home folder. Now type "bash" to go to the bash shell. After that, source the file by typing ". <space> ~/syn_setup", where <space> is a space character. You are done to run HSIM and HSPICE now.

Chapter 2

Modifying the HSIM and HSPICE Path in the Setup file

In order to select the proper binary for AMD64 bit machines or Intel X86 machines, specify the same by *adding a new export path* or *uncomment the proper export path* statements. Say for example, in the file syn_setup we have two lines to be defined for HSIM path as provided below:

export PATH=$PATH:$HS_HOME/hsimplus/bin

#export PATH=$PATH:$HS_HOME/hsimplus/platform/amd64/bin

If you want to use the amd64 bit binary, you need to uncomment the same, and comment the normal binary which should look like below after doing so:

#export PATH=$PATH:$HS_HOME/hsimplus/bin

export PATH=$PATH:$HS_HOME/hsimplus/platform/amd64/bin

Thus, you can modify or uncomment the proper path for the binary for HSIM or HSPICE.

Chapter 3

Running the HSIM or HSPICE on a SPICE file

The basic command for running HSIM or HSPICE is provided below:

HSIM/HSPICE –i <Input Spice File Name> -o <Output File Name>

You can provide the file name directly without specifying the input and output options. In that case, the output files have a common and fixed name which can prove to be overwriting all the different simulation outputs. Hence, to avoid the conflict with unspecified output file name, you should specify a definite output file name by the above syntax.

Chapter 4

Viewing the Output of *.tr0 and *.fsdb files

In order to view the *.tr0 created by HSPICE or *.fsdb created by HSIM, open the *Custom WaveView* by typing “wv&” in the command prompt. Now select *open folder symbol* to browse through the folders and select a *.tr0 or *.fsdb file and plot necessary signals to analyze the simulation. FYI the *.tr0 can be *.tr1 or *.tr2 or in general *.trx where x is a number like 0, 1, 2 3 etc..

Chapter 5

Important HSIM Parameters

For HSIM simulations one thing should be noted that it is completely different for HSPICE in terms of added sensitive parameters. If somebody does not specify the accurate HSIM parameters one can end up in a completely wrong simulation output. Some of the very important HSIM parameters are provided below:

.param HSIMOUTPUT=fsdb

.param HSIMVECTORFILE=<Vector File Name>

.param HSIMREDEFSUB=1

.param HSIMHIERID='.'

.param HSIMANALOG=0

.param HSIMPOSTL=0

.param HSIMALLOWEDDV='VDD*0.1'

.param HSIMDCITER=100

.param HSIMVDD='VDD'

.param HSIMSPEED=3

It is recommended that one should at least use the above parameters except HSIMVECTORFILE if not needed. Here, the usual setting for HSIMSPEED is "3" which should give almost accurate results in above sub-vt design. Use HSIMSPEED=5 for functionality checking only in simulations which does not requires high accuracy. Use HSIMSPEED=0 for SPICE like highest accuracy. Do not use HSIMSPEED greater than 2 in case of sub-vt simulations. It is recommended to have HSIMSPEED =0 for sub-vt simulations. For above sub-vt simulations HSIMALLOWEDDV should be set to 10% of the VDD. And for sub-vt simulations you should make it more sensitive to even 5% or 1% of the VDD. Kindly refer to HSIM manual for more details of the parameters.

Chapter 6

Writing Measurement file in HSIM and HSPICE

It is extremely important to measure currents, voltages, timing and other parameters in a design. In order to facilitate the measurement process HSIM and HSPICE offers a broad range of support to .measure statements. Measurement results are stored in *.mt files. We will discuss a couple of measurement types here.

Type “when” is specified as

.measure tran <Measured Parameter Name> when v(<Signal Name 1>)=<Voltage Value> td=<Initial Time After the Signal to be Searched> rise/fall=<Number of rise or fall reference>

Example:

*.measure tran tcyc_1 when v(CLK)='VDD*0.1' td=0us rise=1*

Type “average from to”

.meas tran <Measured Parameter Name> avg I(<Signal Name>) from=<Time Stamp 1> to=<Time Stamp 2>

Example:

.meas tran IAvg_Write_1 avg I(VVDDUS) from='tcyc_1' to='tcyc_2'

Type “parameter calculations”

.meas <Measured Parameter Name> param='Calculations goes here'

Examples:

.measure ILeakAvgPeriphery param='(ILeakAvg - ILeakAvgArray)'

*.measure Energy_Wr_1 param='1/2*Cpd_Wr_1*(VDD^2)'*

Kindly refer to the HSPICE/HSIM manual for detail information regarding measurement statements.